IV International Conference on Computational Methods for Coupled Problems in Science and Engineering
COUPLED PROBLEMS 2011
M. Papadrakakis, E. Oñate and B. Schrefler (Eds)

THREE DOMAIN THERMAL AND MECHANICAL FLUID-STRUCTRURE INTERACTION ANALYSIS APPLIED TO COOLED ROCKET THRUST CHAMBERS

DANIEL S. C. KOWOLLIK, MATTHIAS C. HAUPT AND PETER HORST

Institute of Aircraft Design and Lightweight Structures
Technical University Braunschweig
Herrmann-Blenk-Str. 35, 38106 Braunschweig, Germany
e-mail: d.kowollik@tu-bs.de, web page: http://www.ifl.tu-bs.de

Key words: FSI, CFD, TBC, Coupling, Thrust Chamber, Liquid Rocket Engine

Abstract. Regeneratively cooled combustion chamber and nozzle structures are exposed to extreme temperature gradients in space and time. One sided wall heating during the hot run generates thermomechanical loads that induce high heat fluxes on the surface and consequently high stresses inside the thin cooling channel structures. In order to address the strong interaction between the structure and the different flow fields a coupled simulation considering the thermal and mechanical interactions is desirable. The present study covers both physical couplings in a partitioned approach applied to the steady state case of a subscale thrust chamber.

Furthermore, this study will present a novel full parametric 3D modelling approach for cooled rocket thrust chambers, which is specifically designed to reduce computational expense in FSI analyses by applying non conforming symmetry conditions across coupling interfaces. The parametrization ranges from thrust chamber contour design through CAD modelling up to grid generation of the individual domains. Further extension of the parametric approach allows the analysis of thermal protection systems applied inside the combustion chamber.

1 INTRODUCTION

The regeneratively cooled rocket thrust chamber is one of the most important components determining the performance of today's launch vehicles. Over the years research has been performed with the goal to optimize the overall performance of rocket engines. Complex thermal and mechanichal loading conditions cause a strong interaction between the hot combustion gas, the thin wall structure of the combustion chamber and the cryogenic liquid fuel which is pumped through the high aspect ratio cooling channels. Combustion gases reach a temperature level of about 3,600 K with a chamber pressure up to 20 MPa [1]. The highest heat flux is reached with 131–147 MW/m^2 at the throat region [2].

The response of the cooling channel structure can only be addressed through an adequate prediction of the in-service conditions. Chemical reactions during the combustion process, fluid flow in the thrust chamber and cooling channels, heat transfer between the involved domains and the thermomechanical coupling in the structure make up a complex system. The thermal and pressure loads generated by the hot gas and cooling channel flow are essential for the prediction of the cooling channel response.

Computer resources in the past were limited to the application of simplified models for coupled simulations. In [3] the thermal interaction between a 2D structural model and a straight 3D cooling channel segment is analysed. The interaction with the hot gas side is neglected, whereas experimental data of [4] serve as boundary condition. This simplified model is extended by a thermomechanical coupling strategy in [5]. Riccius argues in [6] about the relevance of a correct transient load cycle, whereas a 1D fluid model is deployed for quasistatic and transient thermal analyses. A 3D coupled heat transfer analysis is presented by Liu et al. in [7], where the hot gas and cooling channel structure are modelled by a finite volume scheme. The cooling channel flow is included by a 1D approach. Current industrial simulation strategies were presented by Knab et al. in [8], where a coupled heat transfer simulation is performed between the hot gas side and the coolant flow. The hot gas side is modelled by an axisymmetric multi-phase Navier-Stokes solver, whereas the cooling channel flow and the structure are analysed by a 3D conjugate heat transfer model.

The increasing computing capacity now allows us to perform a coupled 3D fluid-structure interaction analysis of the whole thrust chamber structure. A coupled steady state heat transfer analysis with a subsequent static mechanical analysis of a subscale LOX/H_2 thrust chamber is investigated in this work.

Assuming finalized combustion at the inlet, the hot gas flow is determined by means of the 3D steady RANS solver of the CFD code DLR-TAU [9]. ABAQUS FE Software is used to analyse the cooling channel response in a hot gas run at steady state.

The FSI computation environment ifls developed at the Institute of Aircraft Design and Lightweight Structure at the Technical University of Braunschweig will be shown to be able to perform the fully coupled 3D FSI Analysis of the thrust chamber structure.

Building parametric models for FSI Analysis is a laborious work, because each physical domain has to be carefully designed starting from CAD modelling, going through specific grid generation techniques according to the physics involved and finally defining the appropriate boundary conditions, where e.g. discretised coupling surfaces have to be extracted for interpolation purposes of the involved field and flux variables considering non-conforming grids. In this study we will present a novel object oriented full parametric 3D modelling approach for cooled rocket thrust chambers. The approach covers all above mentioned modelling steps in one software concept.

In the following sections we will present the partitioned numerical coupling concept implemented in ifls, outline the parametric modelling approach and demonstrate the complete coupling approach of this study on a steady state subscale thrust chamber computation, which will be discussed.

2 NUMERICAL FSI COUPLING CONCEPT

The developed simulation environment ifls provides a framework to simulate coupled problems in a partitioned approach, where the coupling domains are analysed with individual codes for the aerodynamic and structural phenomena. Detailed information of the software concept implemented in ifls and the available state-of-the-art techniques for numerical coupling can be taken from [10]. The software environment has been tested and validated for different individual code combinations. For details, refer to the previous studies [10, 11, 12, 13].

Basic requirement for such a coupled concept is the equilibrium of energy transferred across the coupling surfaces during a single coupling time step. In the following analysis a steady state problem is solved. To ensure conservation in space, non-conforming grids need to be handled such that the transfer of the state variables is achieved in a conservative manner. Several techniques to exchange the boundary conditions on the coupling surface were proposed. A general approach to construct conservative transfer schemes is provided in [14, 15, 16], which fulfill the interface conditions in a weak formulation based on the Lagrangian multiplier technique. Based on this approach ifls provides several popular transfer schemes presented in [10, 17]. The transfer scheme applied in the present study is referred to as conservative interpolation or node projection scheme. In this case the nodal loads of the fluid interface are mapped to the nodes of the closest structural interface element, where the structural shape functions are evaluated at the fluid interface nodes. A schematic illustration of ifls iteration procedure and framework is shown in Fig. 1.

The fluid-structure interaction considered in this work is assumed to be steady state for all involved domains. A two way coupled formulation accounts for the heat transfer and stress/displacement problem between the hot gas and the cooling channel structure. In each euqilibrium iteration step the structural domain is solved in a sequentially coupled thermomechanical analysis, in which first the heat flux computed on the hot gas side serves as input for the structural thermal analysis and second the structurally computed temperature distribution together with the hot gas pressure loads is applied in a subsequent static stress/displacement analysis. The deformation results of the structural analysis are transferred to the fluid domain in order to perform a grid deformation in each iteration step. Furthermore, the surface temperatures computed by the structural solver are interpolated to the fluid side serving as new input in the equilibrium algorithm.

The solution of the coupled problem is obtained by the Dirichlet-Neumann iteration. Formulating the Dirichlet problem in terms of a Schur complement one defines symbolically the fluid operator $\mathcal{F}$ as follows:

$$\mathcal{F}\,\phi^{(f)}|_\Gamma = \phi^{(f)}_{,\mathbf{n}}|_\Gamma \,, \tag{1}$$

where ϕ represents the state variables (temperature, displacements) and $\phi_{,\mathbf{n}}$ the flux (heat flux, surface tractions) at the coupling interface Γ. The indices (s) and (f) at the state

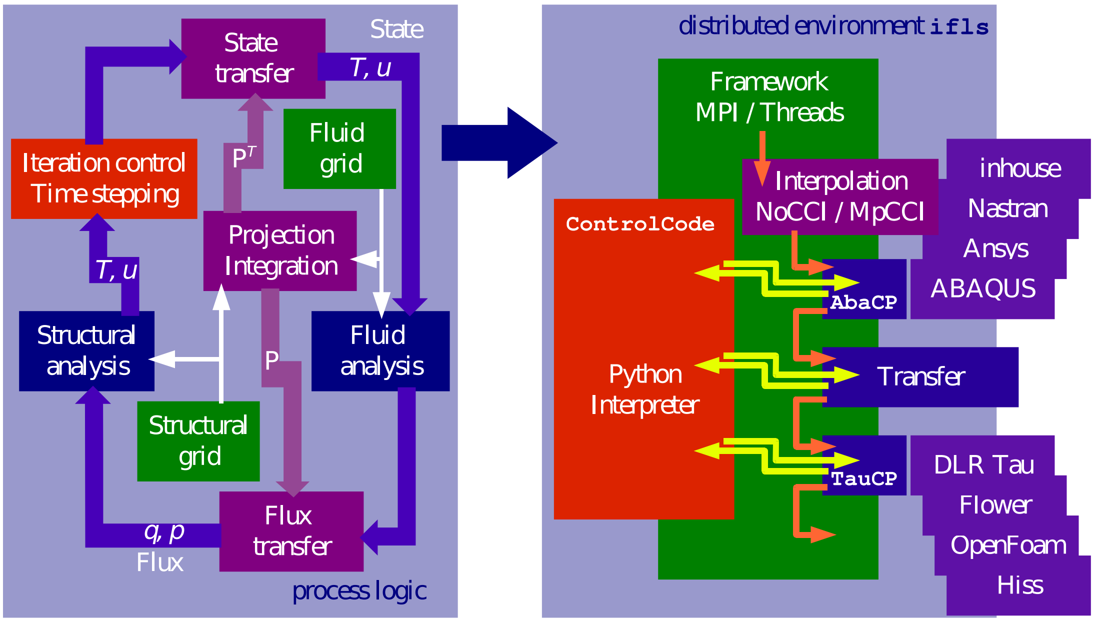


Figure 1: Process logic of the coupling algorithms and the corresponding software architecture

variables ϕ and the flux $\phi_{,\mathbf{n}}$ represent the fluid and the structural domains, respectively. The inverse Schur complement is formulated for the structural problem, consequently the Neumann problem is solved:

$$\phi_{,\mathbf{n}}^{(s)}|_\Gamma = \mathcal{S}^{-1}\,\phi^{(s)}|_\Gamma \;\longleftrightarrow\; \mathcal{S}\,\phi^{(s)}|_\Gamma = \phi_{,\mathbf{n}}^{(s)}|_\Gamma\,. \tag{2}$$

For the classical Dirichlet-Neumann iteration the state variables are relaxed in each steady state iteration step as follows:

$$\phi_{k+1} = (1-\omega)\,\phi_k + \omega\,\mathcal{S}^{-1}\,\mathcal{F}\,\phi_k\,, \tag{3}$$

with the relaxation parameter ω.

The application of a fixed ω between 0.7 and 1.0 can be assumed as standard in coupled heat transfer problems. However, an optimal computed $\omega = \omega_{opt}$ is desirable to accelerate the equilibrium iteration. Several acceleration methods, e.g. Aitken method and gradient method have been analysed in the past [18, 19].

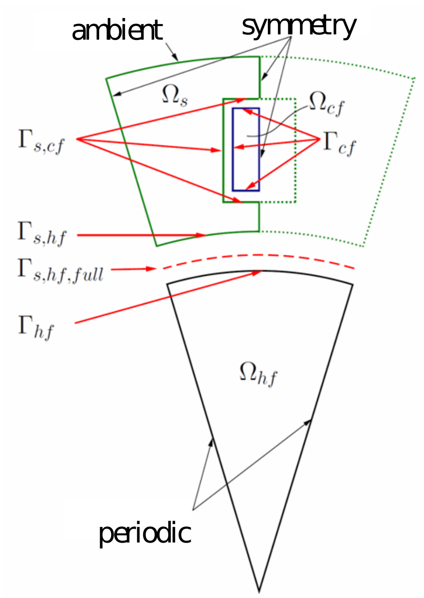
For the fully coupled problem in the present work the Aitken method is used to accelerate the fixed point iteration. The Aitken method is defined as follows:

$$\omega_{opt,k} = 1 - \mu_k\,, \tag{4}$$

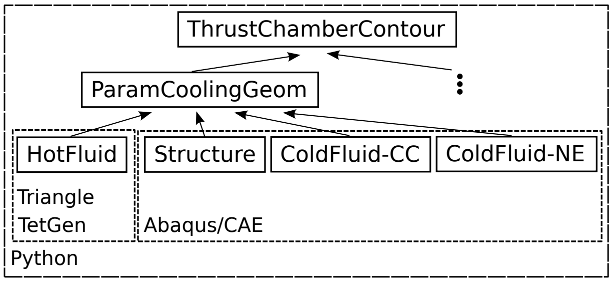
where the Aitken coefficient μ_k is defined as

$$\mu_k = \mu_{k-1} + (\mu_{k-1} - 1)\frac{(\Delta\phi_k - \Delta\phi_{k+1})^{\mathrm{T}}\,\Delta\phi_{k+1}}{(\Delta\phi_k - \Delta\phi_{k+1})^2} \quad \text{for } \mu_k \geq 1, \tag{5}$$

with $\Delta\phi_{k+1} = \phi_k - \phi_{k+1}$. For the first iteration loop $\mu_k = 0$.



(a) Definition of coupling surfaces.



(b) Software concept.

Figure 2: Parametrized modelling aspects of cooled rocket engines.

3 PARAMETRIC 3D MODELLING APPROACH

3D FSI analyses of cooled rocket thrust chambers are computationally expensive. Computational reduction can be achieved if symmetry conditions are used, consequently the assumption of the present parametric approach is to model half of a cooling channel segment for the combustion chamber cooling circuit. The hot gas is modelled with periodic boundary conditions, which is shown in Fig. 2(a), where the coupling surfaces are defined. The data transfer between the hot gas and structural domain is achieved through an additional coupling surface $\Gamma_{s,hf,full}$, where the state and flux variables are mirrored to satisfy integrity of the applied boundary conditions in each iteration step. Accounting for all symmetry conditions in the different computational domains allows the simulation of the 3D state by the assumption of periodic repetition.

In numerical coupling it is very time consuming to realize CAD modelling and grid generation for all involved computational domains. Validation of numerical methods is always a topic, therefore a software concept for parametrized modelling of thrust chamber designs and cooling channel setups was developed. The parametrization reaches from the thrust chamber contour and cooling channel design to the CAD modelling and finally to the grid generation of the hot gas, structure and cooling channel flow field.

Fig. 2(b) shows schematically the developed software architecture, which uses the python interface to the preprocessor Abaqus/CAE. Python scripting allows an object oriented modular and reusable framework for the different components of the parametrized models. Additional software packages like Triangle [20] and TetGen [21] enhance

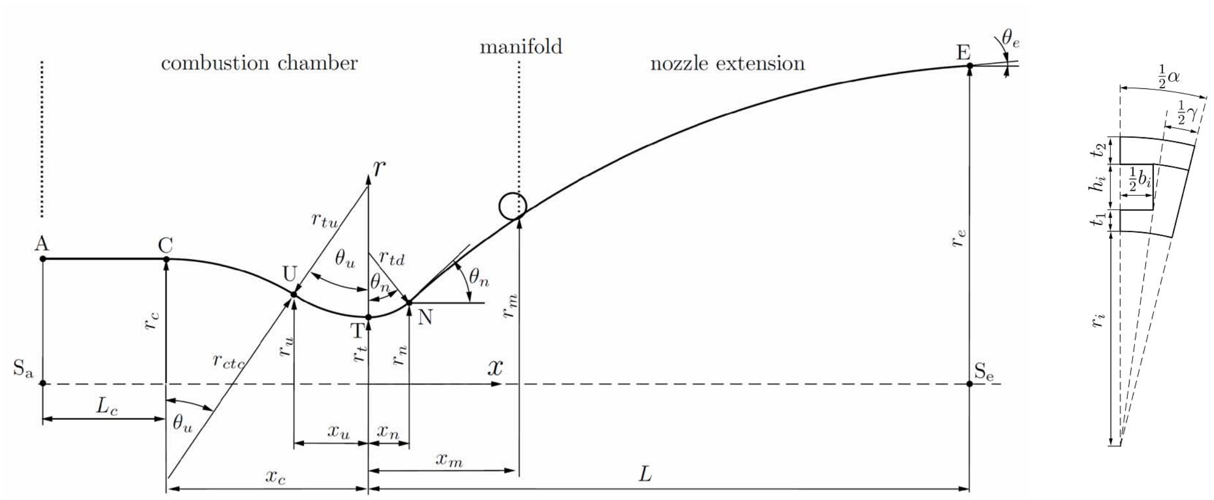


Figure 3: Parametrized model: thrust chamber contour (left); cooling channel cross section (right).

the possibilities in grid generation.

The class definition of *ThrustChamberContour* describes the complete rocket engine contour of Fig. 3 (left), where the nozzle is designed with the thrust optimized parabola (TOP) from Rao [22]:

$$\left(\frac{b\,x}{r_t} + \frac{r}{r_t}\right)^2 + \frac{c\,x}{r_t} + \frac{d\,r}{r_t} + e = 0. \tag{6}$$

Even modern nozzles like the Vulcain and SSME nozzle can be studied by means of a parabolic contour [23]. The nozzle contour can be described by five independent variables r_{td}, θ_n, L, r_e and θ_e. The angles θ_n and θ_e are evaluated through a bivariate spline whose input data was taken from Rao [22].

The derived class *ParamCoolingGeom* accounts for the chosen cooling circuit cross section definition shown on the right of Fig. 3. Different cooling channel setups are possible and can be derived likewise. For each cooling channel setup, one can derive classes for the different computational domains. These classes use the object oriented Abaqus/CAE interface to implement the parametrized CAD and grid models. In this study the analysed subscale thrust chamber consists of three domains structure, hot gas and the two cooling curcuits, one in the combustion chamber region and one in the nozzle extension. For both cooling circuits a continously varying cooling channel geometry can be defined for the cross section sketched in Fig. 3 (right).

4 NUMERICAL RESULTS

4.1 Cooled subscale thrust chamber

In this study a coupled FSI analysis of a LOX/H_2 subscale rocket thrust chamber is simulated. The subscale thrust chamber is defined by Astrium Space Transportation GmbH, Propulsion & Equipment. Extracted general parameters of the detailed test case

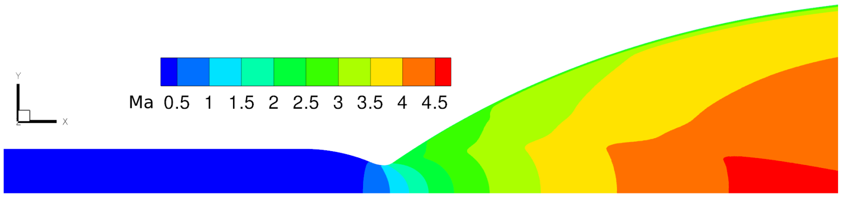


Figure 4: Mach number distribution at steady state of the coupled FSI analysis.

definition serve as input for the parametrized thrust chamber contour of this work. The test case consists of 80 cooling channels in the combustion chamber and 160 cooling channels in the nickel based nozzle extension. The combustion chamber material setup is composed of a NARloy-Z [24] liner and an INCONEL alloy 600 jacket.

On the hot gas side a 3D steady RANS analysis using ideal gas law assuming finalized combustion is performed in each Dirichlet step of the static FSI analysis. For the fluid simulaton the DLR TAU-Code is used, which is an unstructured RANS solver based on the finite volume method. Reservoir pressure inflow conditions were computed with the preliminary design tool RPA (Rocket Propulsion Analysis) [25] and served as inlet conditions for the DLR TAU-Code. The temperature niveau computed with RPA reaches 3502 K at a pressure level of 9.35 MPa. At the outlet supersonic outlet conditions are applied, while an isothermal wall is defined at the coupling surface. The turbulent effects are modelled by the original version of the Spalart-Allmaras model implemented in the TAU-Code. The hybrid grid consists of 590833 grid points, 686535 tetrahedra and 785686 prisms. The dimensionless $y+$ is adapted to a maximum of 0.57 at the thrust chamber throat. Fig. 4 shows the typical mach number profile of a TOP nozzle, where compression waves are generated at the intersection of the downstream arc and the parabola contour. These compression waves coalesce in an internal shock. Comparisons to Östlund approve these flow phenomena [23].

The steady state heat transfer problem of the cooling channel structure is analysed with the ABAQUS FE software. The resultant heatflux of the hot gas simulation is transfered as boundary condition on the coupling surface. In this study the cooling curcuits introduced in section 3 are accounted for by the definition of constant film coefficients at the structural boundary. The film coefficient of $h_{f,CC} = 150\,\mathrm{kW/(m^2\,K)}$ is applied for the combustion chamber circuit and $h_{f,NE} = 30\,\mathrm{kW/(m^2\,K)}$ is applied for the nozzle extension circuit. For both circuits the sink temperature is defined as $T = 40\,\mathrm{K}$. Radiation effects in the combustion chamber are not considered. Radiation to ambient is computed for the outside facing surface of the cooling channel structure.

The resultant temperature distribution of the cooling channel structure is used as input condition for the static stress/displacement analysis. The fluid solver provides the mechanical surface loads for the hot gas side. At the inlet deflections in the axial direction are suppressed. Symmetric deformation is guaranteed by applying zero deflection normal to the symmetry planes sketched in Fig. 2(a). For the static analysis a linear

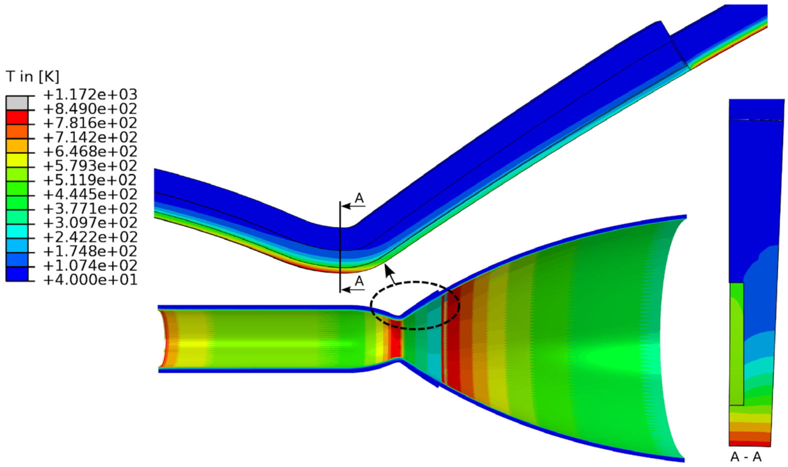


Figure 5: Structural temperature distribution of the cooled rocket engine at steady state.

elastic material model with temperature dependant material parameters is applied for the above mentioned materials. Both finite element models are modelled by 3D 8–node linear hexaeder elements.

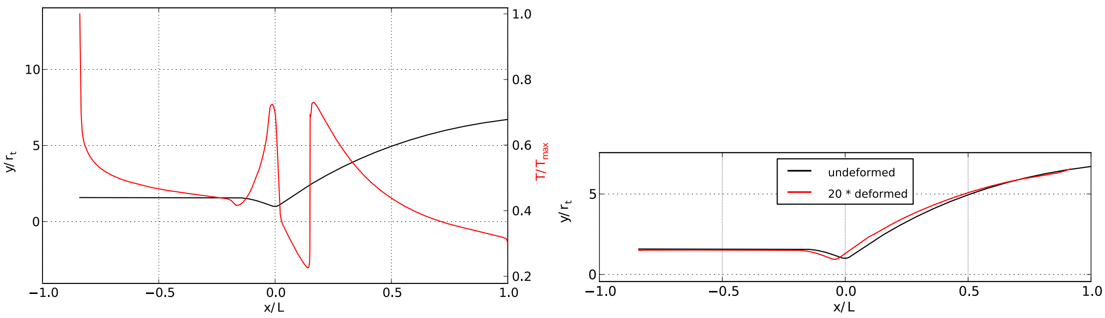
Fig. 5 shows the converged temperature distribution of the structural domain. The high temperatures at the combustion chamber inlet can be explained by the assumption of finalized combustion at the inlet, where a homogenous inflow condition neglecting the typical injector head is applied. Realistic physical conditions are met at the throat region, where the temperature peak reaches about 849 K. The cut view at the throat region shown in Fig. 5 gives insights about the temperature distribution of the cooling channel cross section.

Another temperature peak can be identified in Fig. 6(a) at the nozzle extension just downstream of the manifold position. The nozzle extension material is completely out of INCONEL alloy 600, which has a heat conductivity one order lower than the upstream NARloy-Z. Furthermore, the assumed constant film coefficient is five times higher in the upstream cooling curcuit.

The structural response of the static stress/displacement analysis is shown for the thrust chamber counter in Fig.6(b). The effect of the global deformed state on the aerothermomechanical analysis is small compared to the strong influence of the thermal interactions.

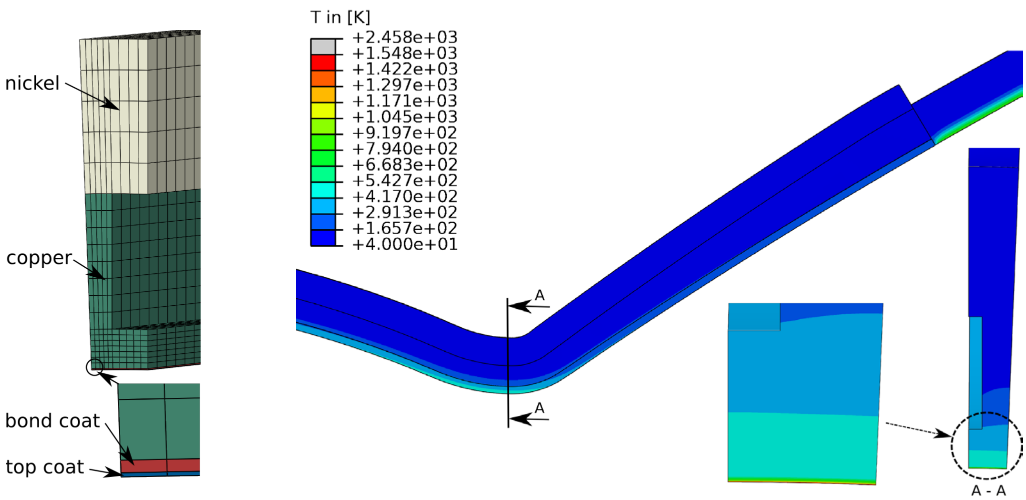
4.2 FSI analysis including thermal protection systems

The transfer of thermal barrier coatings (TBCs) to the high heat flux environment of rocket thrust chambers seems straightforward because TBCs are effectively used in power generation and other aerospace applications. Referring to thrust chambers, TBCs offer great potential to reduce heat flux, coolant temperature and pressure loss; consequently an increased chamber life is expected.



(a) temperature distribution. (b) deformed vs undeformed.

Figure 6: Results along the thrust chamber contour of a static aerothermomechanical analysis.



(a) FEM model. (b) Structural temperature distribution.

Figure 7: The cooled rocket engine at steady state including the TBC–system.

The parametrized modelling approach introduced in section 3 can be easily extended to analyse TBC systems in the context of a coupled thrust chamber analysis. In this study a standard TBC coating system consisting of a 30μm MCrAlY bond coat and a 10μm zirconia top coat is analysed. A small section of the parametrized FEM model is shown in Fig. 7(a). The thin layers of the TBC system are analyzed by 8–node continuum shells in which the copper and nickel alloy is modelled by 8–node solid elements like before. Material parameters of copper, BC and TBC are taken from [26, 27] and [24].

Fig. 7(b) shows the converged temperature distribution near the throat region of the structural domain including the TBC system. The temperature peak near the throat region reaches about 1548K. One can depict from the cut view shown in Fig. 7(b) that the temperature level of the copper alloy is drastically reduced compared to the conventional setup shown in Fig. 5.

5 CONCLUSIONS AND OUTLOOK

- Both physical couplings, thermal and mechanical interactions, have been analysed for a steady state case of a subscale thrust chamber. It was shown that the developed simulation environment ifls is capable of simulating FSI phenomena of cooled rocket thrust chambers.

- A novel parametrized 3D modelling approach for cooled rocket thrust chambers has been introduced. The approach is object oriented and easily extendable, which allows for future sensitivity analyses and quick design changes and most important, to understand in more depth the physical coupling phenomena involved.

- At last, the extensibility of the presented modelling approach was shown by a fully coupled steady state thrust chamber analysis including a TBC system applied inside the combustion chamber.

- In the ongoing research we are focused on the transient analysis of complete engine cycles at in-service conditions in order to address the limiting lifetime factors of cooled combustion chambers. In this context the cooling fluid domains analysed by RANS model will be integrated in the presented fully coupled approach.

Acknowledgments

Financial support has been provided by the German Science Foundation (Deutsche Forschungsgemeinschaft - DFG) in the framework of the Collaborative Research Centre Transregio 40. The authors would like to thank Astrium Space Transportation GmbH, Propulsion & Equipment for defining the subscale thrust chamber, which was studied in the present work.

REFERENCES

[1] Sung, I.-K. and Anderson, W., A subscale-based rocket combustor life prediction Methodology. *AIAA, 2005–3570, 41st Joint Propulsion Conference* (2005).

[2] Quentmeyer, R.J., Experimental fatigue life investigation of cylindrical thrust chambers. *AIAA, 77–893 13th Propulsion Conference* (1977).

[3] Kuhl, D., Woschnak, A. and Haidn ,O.J., Coupled heat transfer and stress analysis of rocket combustion chambers. *AIAA, 98–3373, 34th Joint Propulsion Conference* (1998).

[4] Fischer, S.C., Popp, M. and Quentmeyer, R.J., Thrust chamber cooling and heat transfer. *2nd International Symposium on Liquid Rocket Propulsion* (1995).

[5] Kuhl, D., Riccius, J.R. and Haidn, O.J., Thermomechanical fluid-structure analysis and mathematical optimization techniques applied to cryogenic liquid rocket engine design. *4th International Symposium on Liquid Space Propulsion* (2000).

[6] Riccius, J.R., Haidn, O.J. and Zametaev, E.B., Influence of time dependent effects on the estimated life time of liquid rocket combustion chamber walls. *AIAA, 2004–3670, 40th Joint Propulsion Conference* (2004).

[7] Liu, Q.Y., Luke, E.A. and Cinella, P., Coupling heat transfer and fluid flow solvers for multidisciplinary simulations. *Journal of Thermophysics and Heat Transfer* (2005) **19**(4), 417–427.

[8] Knab, O., Frey, M., Görgen, J., Maeding, C., Quering, K. and Wiedmann, D., Progress in combustion and heat transfer modelling in rocket thrust chamber applied engineering. *AIAA, 2009–5477, 45th Joint Propulsion Conference* (2009).

[9] TAU-Code DLR, Technical documentation of the DLR TAU-Code. *Institut of Aerodynamics and Flow Technology Braunschweig* (2009).

[10] Haupt, M.C., Niesner, R. and Unger, R., Computational Aero-Structural Coupling for Hypersonic Applications. *AIAA, 2006–3252, 9th Joint Thermophysics and Heat Transfer Conference* (2006).

[11] Haupt, M.C., Niesner, R., Unger R. and Horst, P., Numerical Analysis of Thin-Walled Structures in Hypersonic Flow Regime. *ICAS 2008 Congress* (2008).

[12] Haupt, M.C., Niesner, R., Horst, P., Esser, B. and Gülhan, A., Computational Analysis of a Heat Transfer Experiment Including Thermally Induced Deformations. *First Int. Conference on Comp. Meth. for Thermal Problems (ThermaComp2009)* (2009).

[13] Haupt, M.C., Niesner, R. and Horst, P., Thermal-Mechanical Analysis of Functionally Graded Plates Subjected to Hypersonic Flow Conditions. *8th Int. Congress on Thermal Stresses* (2009).

[14] Park, K. and Felippa, C., A Variational Principle for the Formulations of Partitioned Structural Systems. *Int. J. Numer. Meth. Engng.* (2000) **47**, 395–418.

[15] Farhat, C., Lesionne, M. and Tallec, P.L., Load and Motion Transfer Algorithms for Fluid/Structure Interaction Problems with Non-Matching Discrete Interfaces: Momentum and Energy Conservation, Optimal Discretization and Application to Aeroelasticity. *Comp. Meth. Appl. Mech. Engng.* (1998) **157**, 95–114.

[16] Park, K., Felippa, C. and Ohayon, R., Partitioned formulation of internal fluid-structure interaction problems by localized Lagrange multipliers. *Comp. Meth. Appl. Mech. Engng.* (2001) **190**, 2989–3007.

[17] Unger, R., Haupt, M.C. and Horst, P., Application of Lagrange multipliers for coupled problems in fluid and structure interactions. *Computers and Structures* (2007) **85**, 796–809.

[18] Mok, D., Wall, W. and Ramm, E., Accelerated iterative substructuring schemes for instationary fluid-structure-interaction. *First MIT Conference on Computational Fluid and Solid Mechanics* (2001) 1325–1328.

[19] Niesner, R., Haupt, M. and Horst, P., Transient analysis methods for hypersonic applications with thermo-mechanical fluid-structure interaction. *3rd European Conference on Computational Mechanics Solids, Structures and Coupled Problems in Engineering* (2006).

[20] Shewchuk, J., R., Delaunay refinement algorithms for triangular mesh generation. *Computational Geometry: Theory and Applications* (2002) **22**(1-3), 21–74.

[21] Si, H., TetGen: A quality tetrahedral mesh generator and a 3d delaunay triangulator. *http://tetgen.berlios.de* (2006).

[22] Rao, G.V.R., Exhaust nozzle contour for optimum thrust. *Jet Propulsion*, (1958) **28**(6).

[23] Östlund, J., Flow processes in rocket engine nozzles with focus on flow separation and side-loads. *Licentiate Thesis, Royal Institute of Technology, Department of Mechanics, Stockholm* (2002).

[24] Esposito, J.J. and Zabora, R.F., Thrust chamber life prediction volume I - mechanical and physical properties of high performance rocket nozzle material. *NASA CR 134806* (1975).

[25] Ponomarenko, A., RPA: Design Tool for Liquid Rocket Engine Analysis. *http://www.lpre.de/resources/software/RPA_en.htm* (2009).

[26] Deutsches Kupferinstitut, Kupferdatenblatt CuCr1Zr. (2005).

[27] Freborg, A., M., Ferguson, B., L., Brindleym, W., J. and Petrus, G., J., Modeling oxidation induced stresses in thermal barrier coatings. *Material Science and Engineering A* (1998) **245**, 182–190.